



## PCB Creation with Eagle for Beginners

by [killersquiere11](#) on August 21, 2012

### Table of Contents

PCB Creation with Eagle for Beginners .....	1
Intro: PCB Creation with Eagle for Beginners .....	2
Step 1: A Quick Note on How Eagle Works .....	2
Step 2: The Control Panel .....	2
File Downloads .....	3
Step 3: Schematic Window .....	3
Step 4: Board Window .....	4
Step 5: Create a New Project and Schematic .....	5
Step 6: Add the Parts to the Schematic .....	6
Step 7: Connect the Parts .....	6
Step 8: Label and Name all of the Nets .....	7
Step 9: Give the Parts Some Values .....	7
Step 10: Electrical Rule Check (Idiot Check) .....	8
Step 11: Board Layout .....	8
Step 12: Board Layout 2 - Getting on the right side .....	9
Step 13: The Ground Plane .....	10
Step 14: Route the Parts .....	11
Step 15: Thermals and Orphans, revisited .....	12
Step 16: The Design Rule Check .....	13
Step 17: DRC Results .....	15
Step 18: You're done .....	15
Step 19: Questions? Comments? Suggested Improvements? .....	15
Related Instructables .....	16

## Intro: PCB Creation with Eagle for Beginners

Eagle is one of several PCB layout programs that you can get for free (other programs include KiCad and DipTrace). The free version of Eagle is somewhat limited in what it can do, DipTrace slightly more so. KiCad is open-source, and hence is completely free.

I use Eagle because its limitations are reasonable for what I need to do, and I believe that it has a better interface than KiCad.

If you're just installing Eagle, you probably will want to use the 'Run as Freeware' licensing option when it comes up. Note that for this instructable, I'm assuming that you have Eagle 6.1 or higher installed. The files that I upload are stored in Eagle 6's xml format, and as such can't be opened by earlier versions of eagle.

First we'll cover moving around a finished project, then we'll start from scratch and design a board from start to finish.



## Step 1: A Quick Note on How Eagle Works.

Eagle's UI is designed with what is called a modal interface. That is, you select one mode, perform it a bunch of times, as opposed to selecting an object and applying an single operation at a time. When used properly, this allows you to work very rapidly, but it can also be a major source of aggravation if you are used to the Windows-y way of doing things.

## Step 2: The Control Panel

The control panel is the main window of Eagle. When you close it, all windows that it opened get closed as well.

A description of the various categories in the Control Panel:

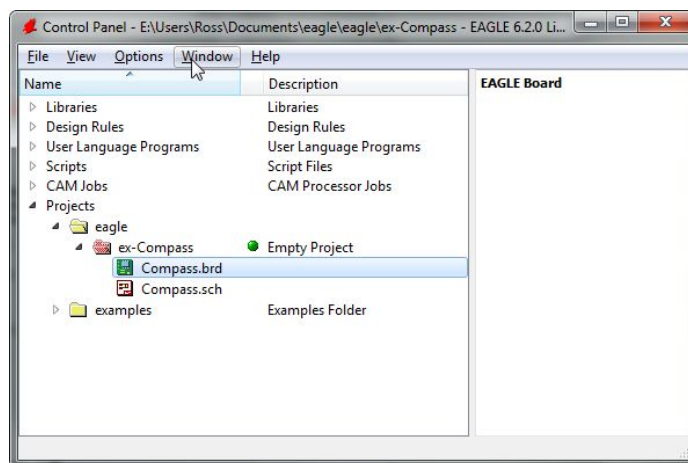
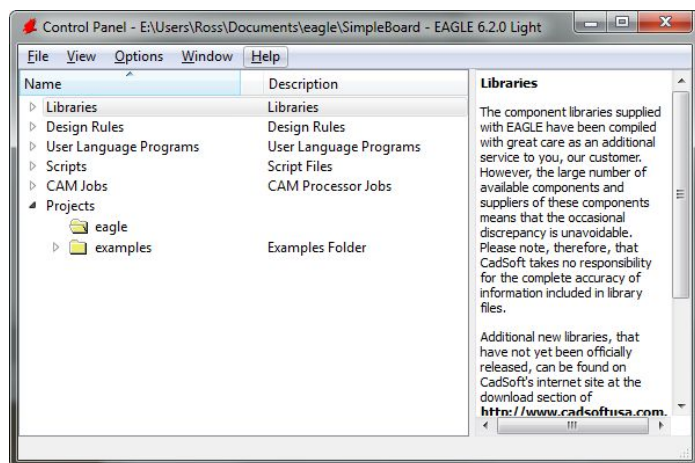
- Libraries (.lbr files) store the individual parts that you add to your board.
- Design Rules (.dru) are what the design rule checker (aka the idiot-checker) uses.
- User Language Programs (.ulp) use Eagle's User Language (pretty much an advanced scripting language) to do stuff that would normally be difficult, tedious, or impossible without them.
- Scripts (.scr) are essentially just groups of eagle commands commands. Simpler and less powerful than ulps.
- CAM Jobs (.cam) export to other formats
- Projects (.sch, .pcb, etc) are where your circuit boards, schematics, and whatever else live

So to start things off, right-click on the 'eagle', select 'New Project', and call it ex-Compass.

Download the 'Compass.sch' and 'Compass.pcb' files from below, and save them in the Documents\eagle\ex-Compass

Hit F5 or go to View-Refresh in the Control Panel to make the files show up there.

Double-click on the Compass.sch file and the schematic and pcb file should both load.



## File Downloads



**Compass.brd** (104 KB)

[NOTE: When saving, if you see .tmp as the file ext, rename it to 'Compass.brd']



**Compass.sch** (224 KB)

[NOTE: When saving, if you see .tmp as the file ext, rename it to 'Compass.sch']

## Step 3: Schematic Window

The schematic window is where you create and edit the schematic (obviously). To start out, try moving around.

There's three different ways to move around in Eagle (If you have a mouse, use 1, otherwise if you have a keyboard that has F# keys, use those, otherwise, if you're some poor soul who doesn't have either, you may be stuck using the scroll bars):

- 1) Use a mouse with a middle-button
  - Middle-click and drag to move around, scroll up and down to zoom in and out
- 2) Use the F# keys
  - F3 is zoom in
  - F4 is zoom out
  - F5 centers the screen where the mouse is.
- 3) Drag the bars on the bottom and sides of the screen and zoom using the button of the top

On the left, just right of all of the buttons, you can see a sheets area. With complex schematics (and non-freeware versions of Eagle), you can create multi-paged schematics and switch between them here. If you are only running the freeware version, you can just close that section and never think of it again...

After you look at the schematic for a while, you may realize that this schematic could be laid out better (especially around the voltage regulator, highlighted above).

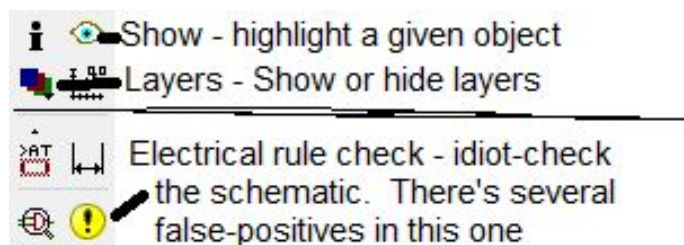
If you click the show button, then click on the VCC net (a net is one of the green lines), you will see that all of the VCC nets get highlighted, including the one connected to that regulator, whose name couldn't be read clearly due to the name of the diode beneath it overlapping with the name of the VCC trace.

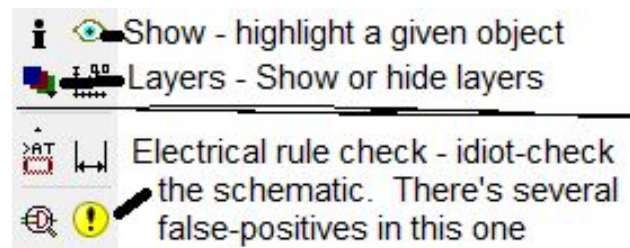
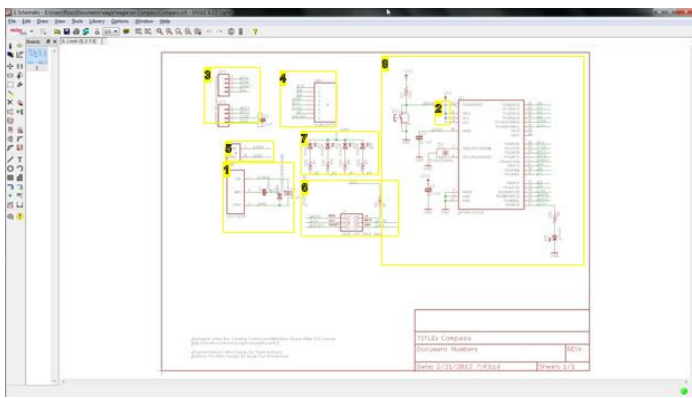
To get the schematic to look cleaner, we can turn off the display of the values of all of the parts. To do this, click the Layers button, and un-check the Values layer (layer 96).

One last way to see any idiosyncrasies that might be missed by the naked eye is the Electrical Rules Check (ERC). This takes what Eagle knows about the various parts (which isn't very much), and checks to see if anything unusual is going on. In the case of this board, there are currently 7 warnings and no errors.

The warnings are thrown because D6 isn't connected to anything, JP1 and JP2 have no values (since they're just through-hole places where I'll wire up the actual compass), GND overlaps another pin since I was trying to get multiple connections to a single pin on one of the parts, but Eagle didn't like that, and since I renamed the 3.3V to VCC, the supply symbols aren't happy. Most of this should be fixed, but, hey, it lets you see what an ERC with warnings looks like...

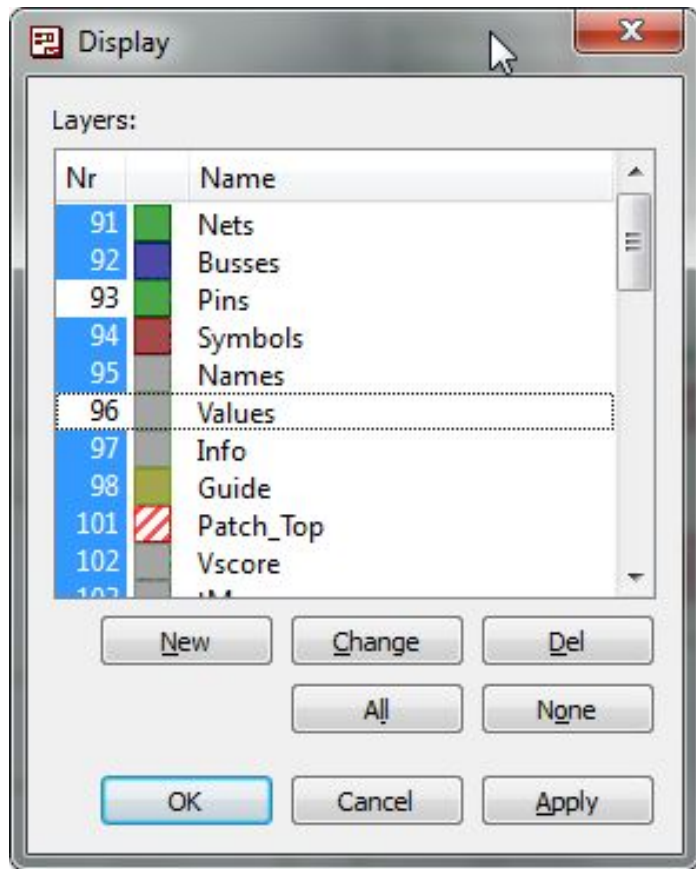
Now click File>Switch to Board to switch to the board layout side of things. If you didn't have a .pcb file paired with this file, it will automatically create one and drop all of the parts outside of the PCB area.





#### Image Notes

1. Voltage Regulator
2. An example VCC net.
3. Compass Module Connector
4. RJ-45 Connector
5. Power Supply
6. Programming Header
7. LEDs
8. Most of an arduino



#### Step 4: Board Window

You move around in the board window just like in the schematic window, so I won't go over that again. Likewise, the Show button is virtually identical.

The Layers button is functionally the same, but will have different layers in this view. Frequently, when looking at a board, you don't need to see all of it at once.

To isolate the top layer, click the Layers button, then click None, then select the following layers:

- Top
- Pads
- Vias
- Unrouted
- Dimension
- tPlace
- tOrigins
- tNames
- tValues (optional, typically I leave this off)

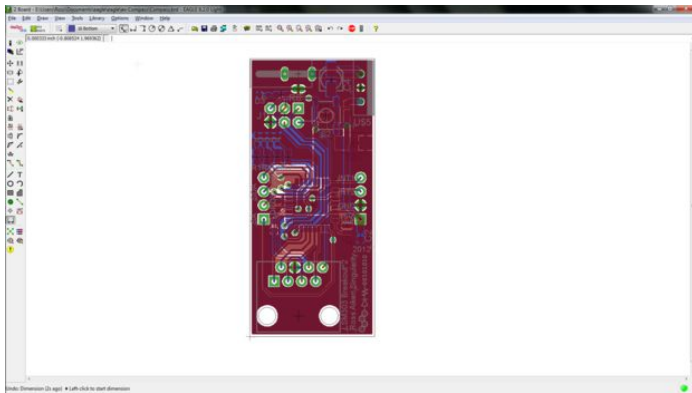
tKeepout (optional, depends on the situation)  
tDocu (optional, adds a lot more clutter)  
\_tsilk

To isolate the bottom layer, click the Layers button, then click None, then select the following layers:

Bottom  
Pads  
Vias  
Unrouted  
Dimension  
bPlace  
bOrigins  
bNames  
bValues (optional, typically I leave this off)  
bKeepout (optional, depends on the situation)  
bDocu (optional, adds a lot more clutter)  
\_bsilk

To view both layers, enable all of the above.

Now that you've seen the very basics of using Eagle, let's plow right ahead into creating your own board.



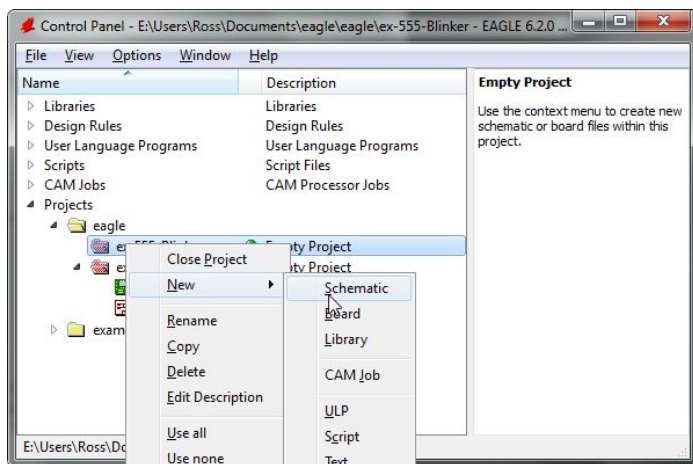
## Step 5: Create a New Project and Schematic

Create a new project by clicking File>New>Project

Give that project a descriptive name (e.g. ex-555-Blinker)

Right-click on the project, and click New>Schematic

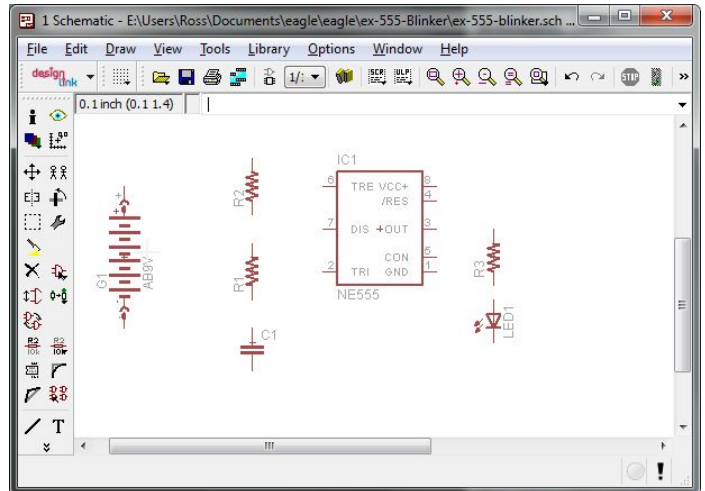
Name that schematic (e.g. ex-555-Blinker.sch))



You'll want the end result to look something like this.

Using this process, add the rest of the parts:

R1, R2, and R3 - 'R0805' from the 'resistor' library  
C1 - 'C0805' from the 'resistor' library (since that makes so much sense)  
LED1 - 'CHIPLED\_0805' part from the 'led' library  
G1 - 'AB9V' part from the 'battery' library



After you've got all of the parts laid out similar to how I showed in the last step, it's time to connect them

Do not use the 'wire' command to make connections like this. 'Wires' are simply cosmetic in the Schematic portion of Eagle, so they won't actually do what you want them to do. If you still do attempt to use the Wire command, I will be forced to unleash my horde of flying robotic monkeys to make sure that you don't try that again.

- It is started with a single-click. Don't click-and-drag or double-click to start.

- After starting, the net will continue being drawn until you single-click on pin or another net, or double-click anywhere.

- Right-clicking will change the way that the net will go from point A to point B. I personally recommend only using the 90-degree-angle forms of this for schematic layout





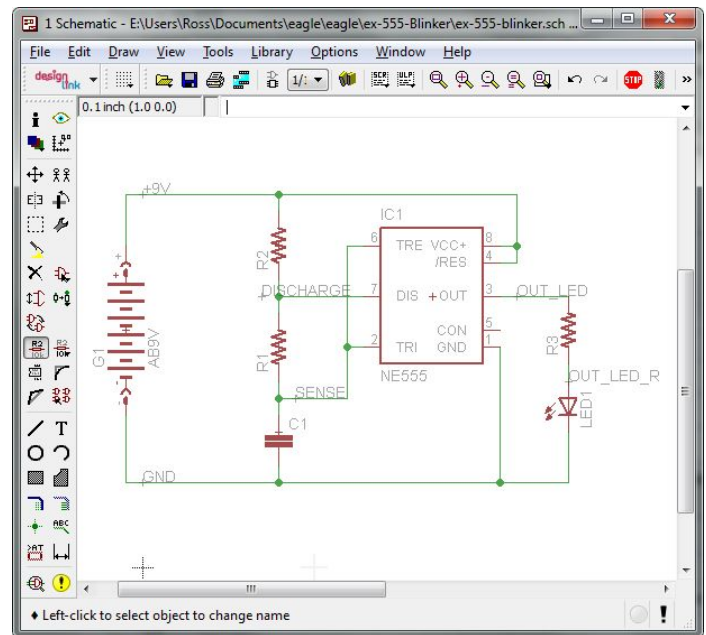
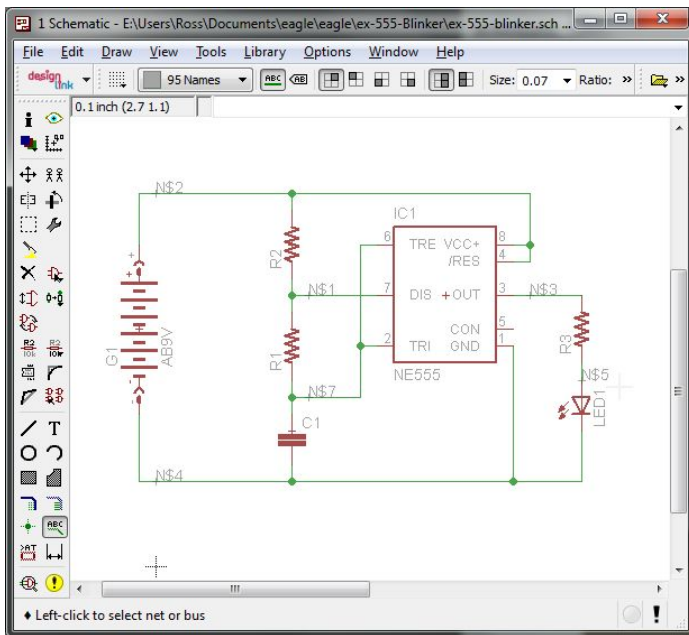
## Step 8: Label and Name all of the Nets

Using the 'Label' command, click on each of the nets to make the nets' names show up on the wires

Once all of the names are showing, use the 'Name' command to give the wires meaningful names, as shown in the second picture in this step.

Labeling the nets is important for two reasons:

- A) It allows anyone who looks at your schematic to at least have an educated guess as to what each portion of it does
- B) When you switch to routing your board, it will be easier for you to tell what each net does and plan accordingly (especially useful when you're dealing with differential traces and whatnot).



## Step 9: Give the Parts Some Values

Using the 'Value' (right next to the 'Name' command), label all of the resistors and the capacitor with the appropriate value

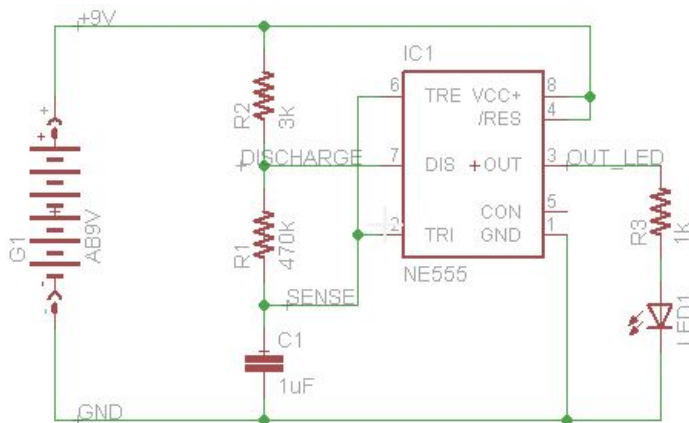
R3 is a current-limiting resistor for the LED, and 1k is a conservative estimate there (depends on the specs of the LED).

C1, R1, and R2 determine the rate of blinkiness of our LED.

According to wikipedia, the frequency of the blinkiness is given by the following formula:

$$f = 1 / (\ln(2) * C * (R1 + R2))$$

So if R1=470k, R2=3k, and C1=1u, that means that this thing will turn on and off 3 times per second



## Step 10: Electrical Rule Check (Idiot Check)

Running the ERC will let you see areas where Eagle thinks you messed up. Let's examine the output for this one line by line...

**Errors (1)** - These warrant you taking a careful look at. Anything here could very well cause your circuit to blow up if you don't pay attention to it.

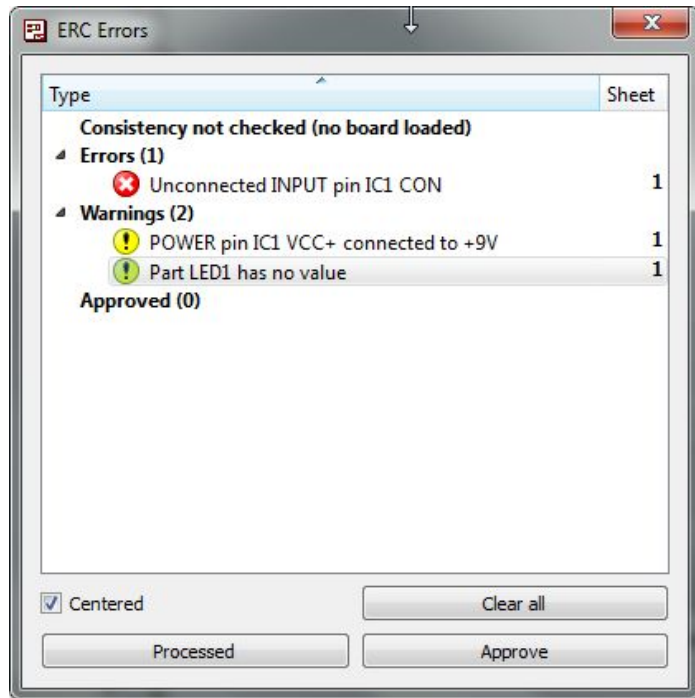
**Unconnected INPUT pin IC1 CON** - In general, unconnected input pins are bad. In this case, the CON pin is a reference voltage that you can manually set, but nothing bad happens if you leave it unconnected (floating).

**Warnings (2)** - These are not as urgent as errors, but still require a cursory glance. One warning to look for is the one about a net only having one node/pin. That means that you didn't connect that net on both ends.

**POWER pin IC1 VCC+ connected to +9V-** Eagle warns you whenever you connect different voltages of power together (if you connect a 12V power supply line and a 5V power supply line, bad things happen). In this case, it's just a nomenclature difference, so it's OK to approve.

**Part LED1 has no value** - If I wasn't as lazy as I am, I'd have given the LED1 part a value, but until then, this warning will exist.

**Approved (0)** - After you click the 'Approve' button on a warning/error, it goes in here.



## Step 11: Board Layout

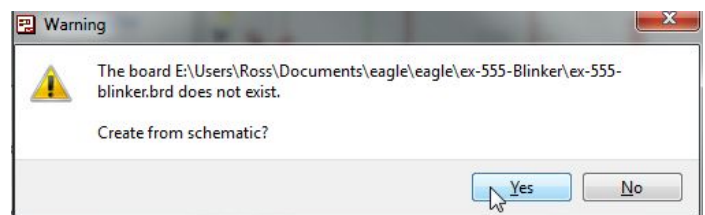
So creating the base of the PCB layout from the schematic is easy. Go to File>Switch to board, and when it warns you that the board doesn't exist, click Yes to create the board from the schematic.

When the board file comes up, there will be a box on the screen, with all of the parts to the left of it. Until you move it, this represents the area where you can place your parts in the free version of eagle. Try to move a part outside of this area and Eagle will yell at you and refuse to cooperate.

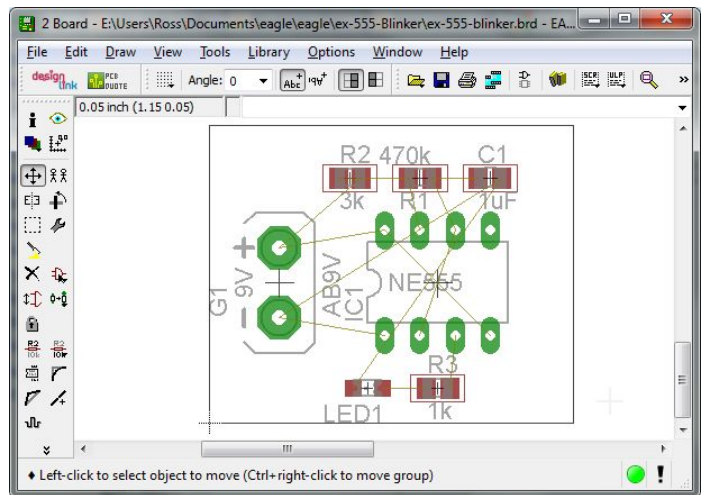
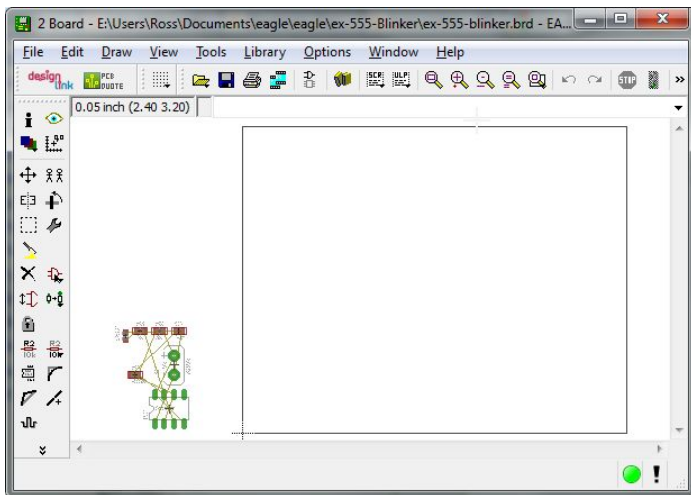
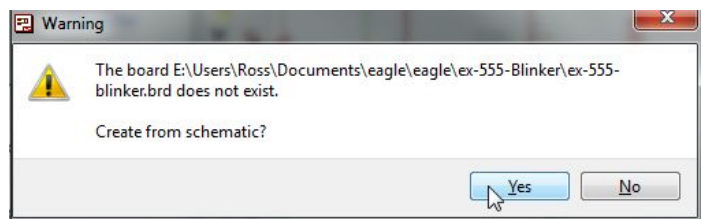
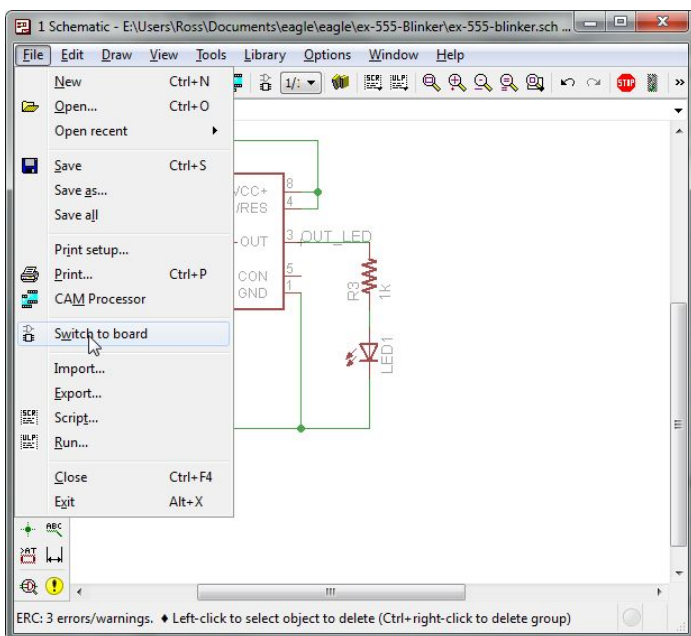
Note that Eagle drops all of the parts that you added into your schematic outside of this placeable area. After you move a part from its resting spot, you have to keep it inside that 4"x3.2" (100x80mm) box.

Move all of the parts into a configuration similar to the one shown in the last picture. Note that this step requires a great deal of forethought to save yourself from headaches later on. Each one of those golden lines represents an unrouted trace.

Typically, when you lay out a board, you first place the parts that have set locations that they need to go, like connectors. Then, group up all parts that logically make sense together, and move these clusters so that they create the smallest amount of crossed unrouted lines. From that point, expand those clusters, moving all of the parts far enough apart that they don't break any design rules and have a minimum of unrouted traces crossing.





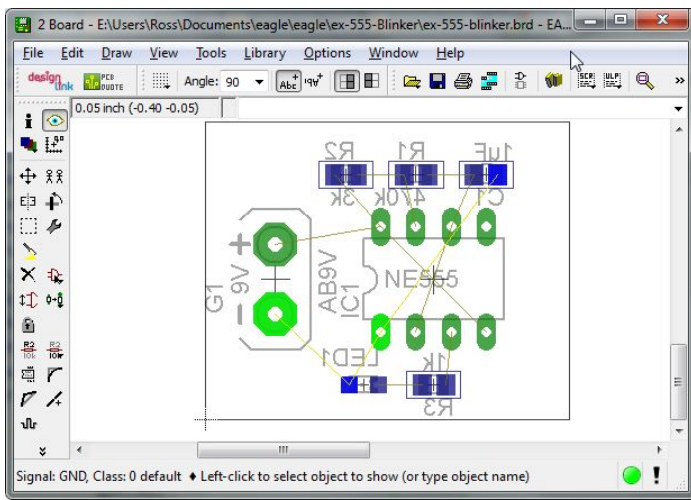


## Step 12: Board Layout 2 - Getting on the right side

One thing with printed circuit boards is that they have two sides. However, you typically pay per layer that you use, and if you are making this board at home, you might only be able to reliably make one-sided boards. Due to the logistics of soldering through-hole parts, this means that we want to use the bottom of the PCB.

Use the Mirror command and click on the surface-mount parts to switch them to the bottom layer. You may need to use the Rotate or Move command to correct the orientation of the parts.

Once you have all of the parts laid out, run the Ratsnest command. Ratsnest recalculates the shortest path for all of the unrouted wires (airwires), which should clear up the clutter on the screen by a fair amount.



### Step 13: The Ground Plane

Ground planes are your friend. They make the remaining steps in this tutorial easier, and they cut back on the time spent etching if you are making the board at home.

Essentially, what a ground plane does is take up all unused space on a board, and connects it to the ground net.

To make a ground plane in Eagle, run the Polygon command. Set the Layer to Bottom (blue), and trace the box around the outside.

Right-click on the blue dashed line and click Properties. A window similar to the one shown in this step should appear..

Here's what each of the boxes mean:

From, To, Length, Angle - Describe the line segment you clicked on

Width - The minimum width (in mils ) of the ground plane

Cap - Not applicable

Curve - Bend of that line segment. Unless you know what you are doing, just leave this at 0

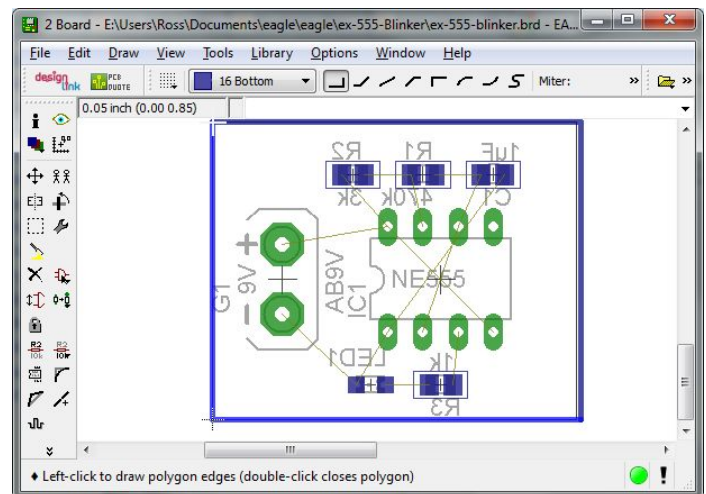
Polygon Pour - Keep this as solid, this affects the pattern.

Spacing - Distance between fill lines when Pour is set to 'Hatch'

Isolate - Distance between the fill and any trace (if this is less than a design rule, it uses the distance specified in the design rule)

Orphans - When this is unchecked, Eagle only fills the largest empty contiguous region. When this is checked, any area that fill can go without violating a design rule will be filled.

Thermals - It's somewhat difficult to describe, just watch a pad/pin on a part and toggle the setting. Essentially, having this enabled makes it much easier to solder parts that are connected to a large plane, at the expense of performance with very high current traces.



**Wire**

From: 0 0.35

To: 0.35 0.35

Length: 0.35

Angle: 0

Width: 0.016

Cap: round

Layer: 16 Bottom

Curve: 0

---

**Polygon**

Polygon Pour: solid

Spacing: 0.05

Isolate: 0

Rank: 1

☐ Orphans

☒ Thermals

---

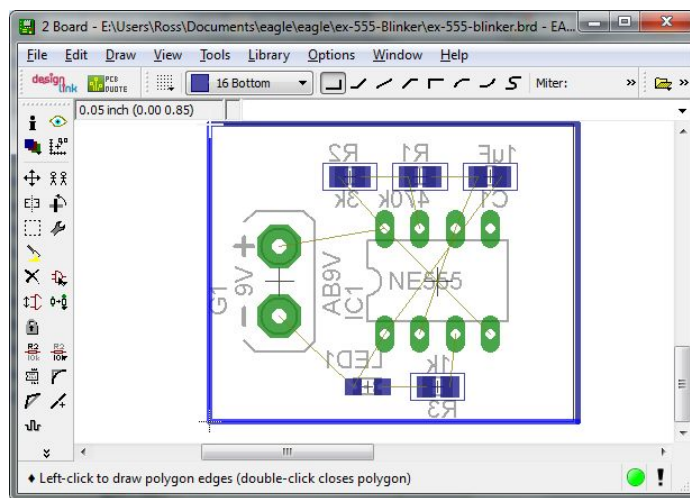
**Signal**

Name: NS1

Net Class: 0 default

☐ Airwires hidden

Apply Cancel OK



## Step 14: Route the Parts

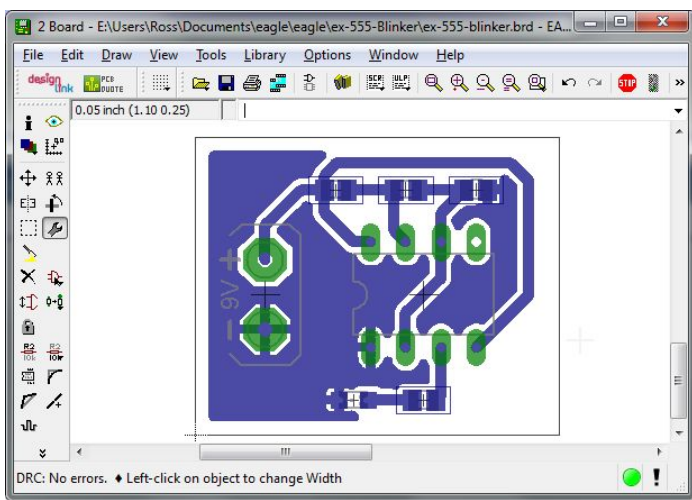
Using the Route command, click on any airwire. A blue trace should appear at the node of the airwire that was closest to where you clicked.

If it is a red trace, that is not what we want. Up top, change the layer from Top (red) to Bottom (blue). Click again somewhere else to anchor the wire at that point. Right-click to change the angle of the turn (ctrl-right-click to switch between turns of the same angle), and middle-click to create a via between the top and bottom layer (shouldn't be necessary or this tutorial).

If the ground plane gets in your way, you can either run Ratsnest to make it recalculate where it goes, or click 'Ripup' then click on the border of the plane. Ripping up the border of the plane will work until you run Ratsnest.

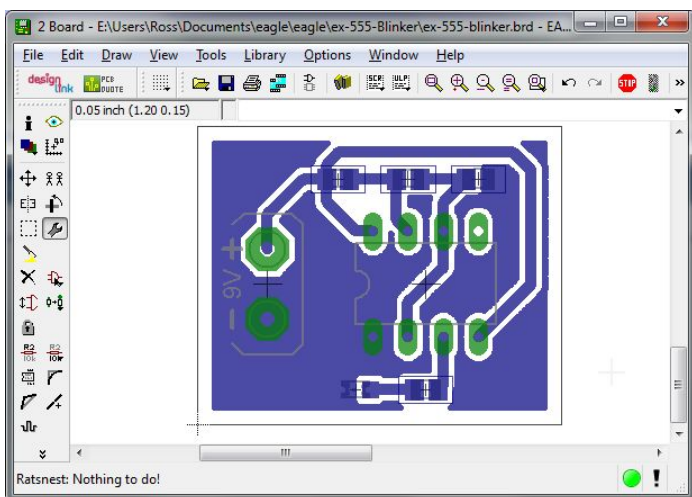
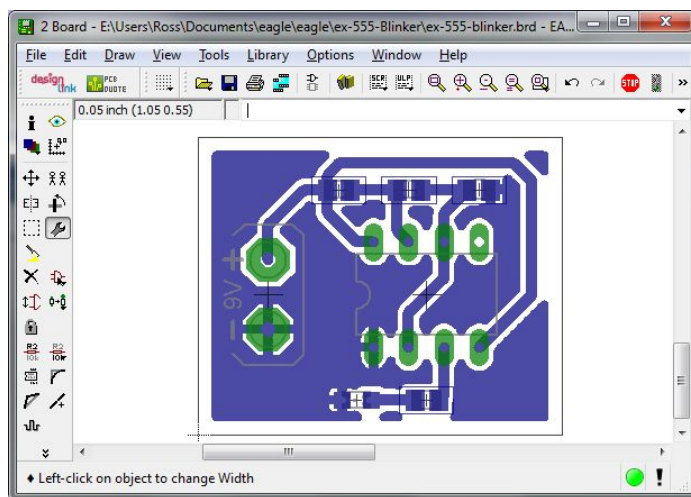
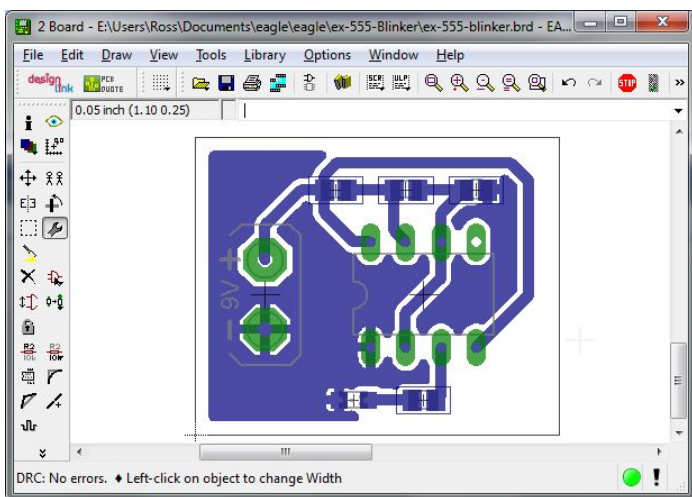
Typically, when routing, we want to use 45-degree angle turns, rather than 90-degree turns like we used in the schematic. With low-speed circuits, this is one of the few concessions that we have to make to physics - sharp turns cause a whole host of funky issues, from electrons leaking from points, to current crowding in the inner corner, to impedance mismatches which cause ringing, reflectance, and a whole host of other issues (all right, some of those are high-speed domain issues, but still, just don't do that).

The example routing that I did is not the only way it could have been done, nor is it the best way, but it is an acceptable outcome.



### Step 15: Thermals and Orphans, revisited

Here are three different screenshots. The first is of the board with thermals and no orphans. The second is thermals and orphans. The third is orphans, but no thermals.



## Step 16: The Design Rule Check

The Design Rule Check (DRC) checks the board you designed against a set of rules to determine if you made any errors. While it isn't perfect, it will catch a large amount of common mistakes.

A quick description of all of the tabs:

**File** - Allows you to pick which DRC file to use. If you are with a group, they might already have one, and services like [OSH Park](#) have a downloadable .drc file that you can Load here.

**Layers** - Since we are using the freeware version of Eagle, you can't really play with this one, but if you have one of the paid licenses, layers can be added by changing Setup to be something like (1\*2\*15\*16)

**Clearance** - This tells Eagle how much room you want between the different types of electrical contacts on the board. If you want, you can everything under Same Signals to 0. OSH Park has a minimum clearance of 6 mils for everything under Different Signals. If you are fabbing the board at home, 20 mils is a reasonable clearance.

**Distance** - Copper/Dimension is the distance from any routing to the edge of the board. Drill/Hole is the distance from any routing to a hole in the board.

**Sizes** - Set the minimum sizes. Minimum Width is another number you pull from your fab house, OSH Park has 6mil, if you're etching it yourself, it should be around 24 mil. Minimum Drill for OSH Park is 13mil, if you're etching it yourself, pick whatever the size of the smallest drill bit you have is. Micro and Blind vias are far beyond the scope of this instructable

**Restring** - Restring controls the size of the through-hole pads/vias. Typically I just leave these at their default values.

**Shapes** - Allows you to make pads have rounded edges. Typically I don't touch this one either.

**Supply** - Remember Thermals from up above? This gives you a bit more control over them. The checkbox allows you to turn on thermals for vias, and Thermal Isolation lets you pick the length of the thermal traces. Typically I don't touch these values at all.

**Masks** - When PCBs are fabricated, they have a coating that covers and insulates all of the board, except for areas designated by the stop mask. After fabrication, if they are being assembled by a machine (or a person with a reflow oven), a stencil is created and solder paste is applied via that stencil to the pads/vias that need to be soldered. Stop controls the size of the openings in the stop mask for the various parts, and Cream controls the size of the openings in the stencil. Once again, the default values work just fine for our typical usage.

**Misc** -

-Check Grid - When you lay out and route the PCB, everything you do is on a grid, whose sizes are determined by the Grid command. If you changed the grid partway through layout/routing, this will ensure that all of your parts obey the new settings.

-Check Angle - If you routed anything with a different style than the 45- or 90-degree turns, or if you moved a part after routing, checking this box will make Eagle yell at you for doing that.

-Check Font

-Check Restrict

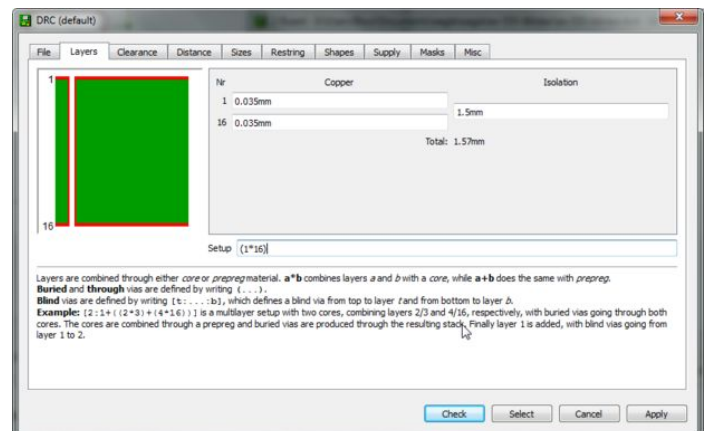
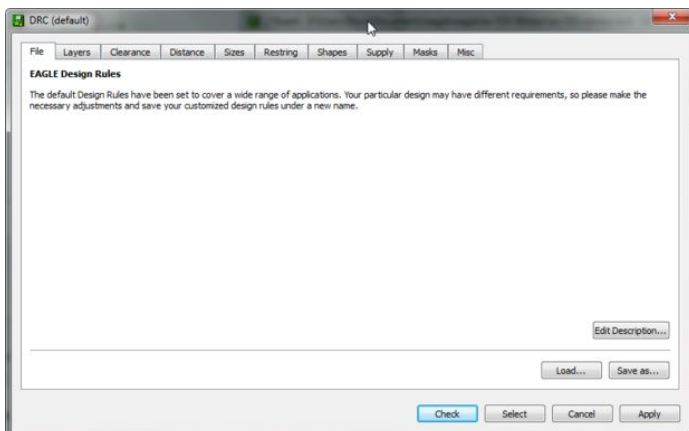
-Max. length difference in differential pairs - Differential routing is where two traces are routed side-by-side and carry a signal that is differential (when wire A is 1, wire B gets set to 0. If A-B is greater than zero, a 1 is being sent, otherwise a 0 is being sent). There are several advantages to this, one of which is what is called 'Common Mode Rejection'. Essentially, most electrical noise more or less adds a voltage to a given wire. If two wires are close enough, the same voltage (let's call it v) gets added to both. With differential routing, we want those two wires to be affected by the same v, so that the v cancels out of (A+v)-(B+v).

Back on track to what this option does, we want the length difference to be minimal so that the wires pick up the same noise. This allows you to pick what difference to choose.

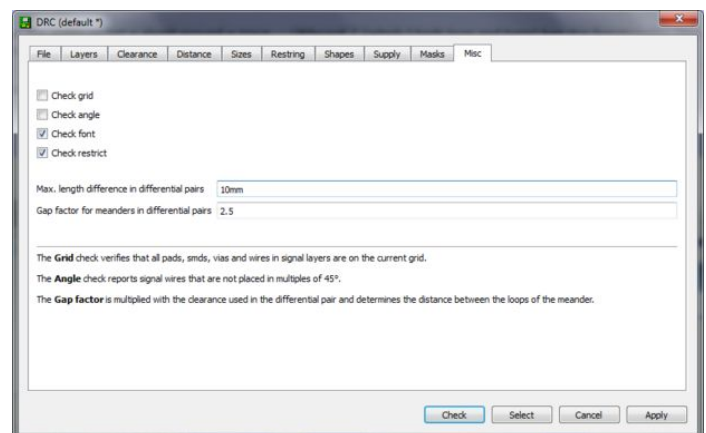
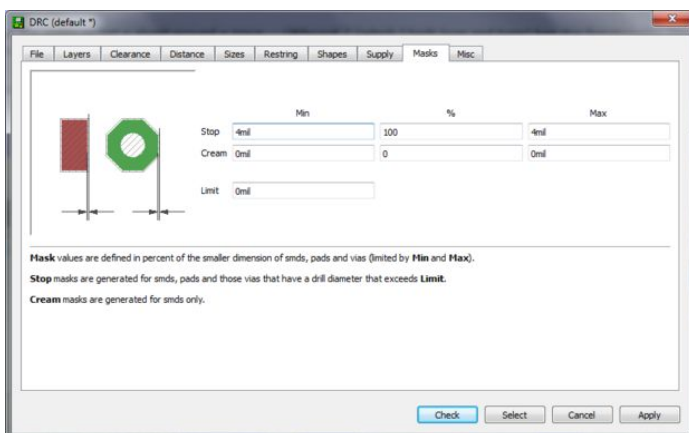
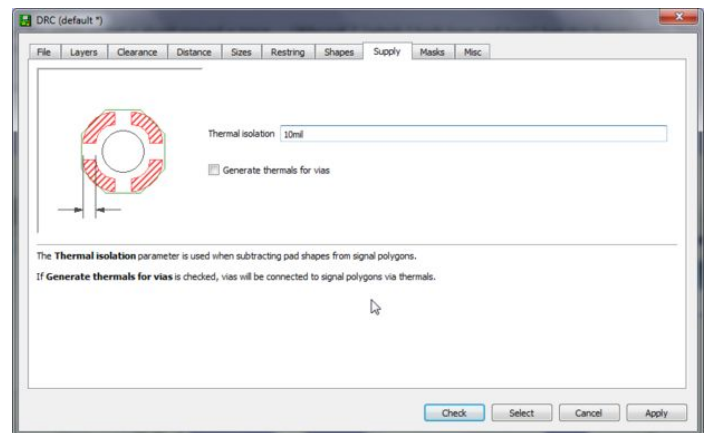
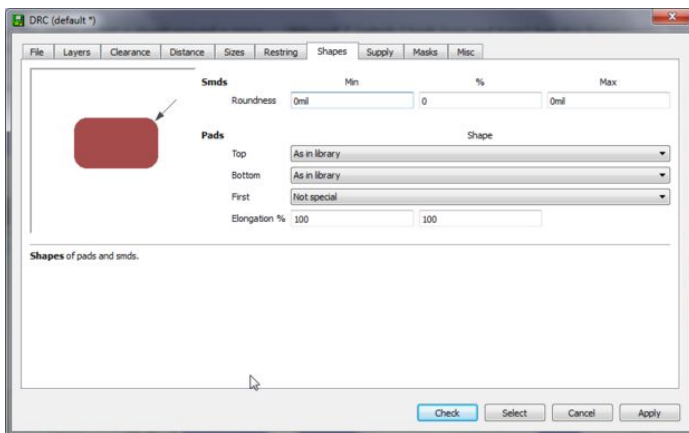
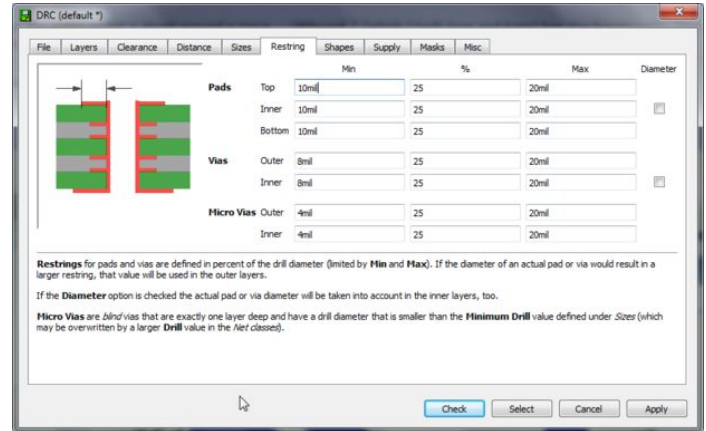
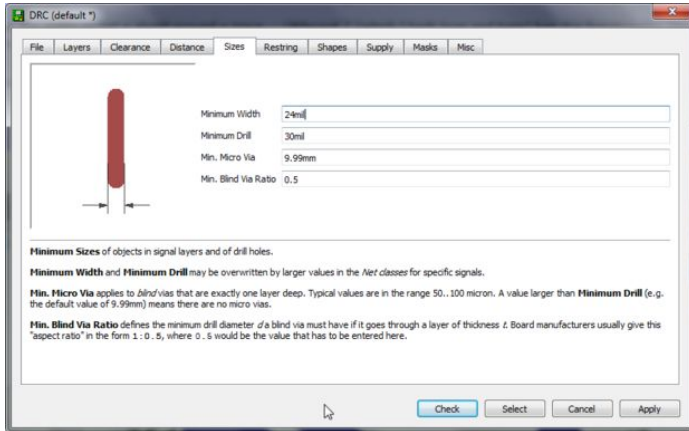
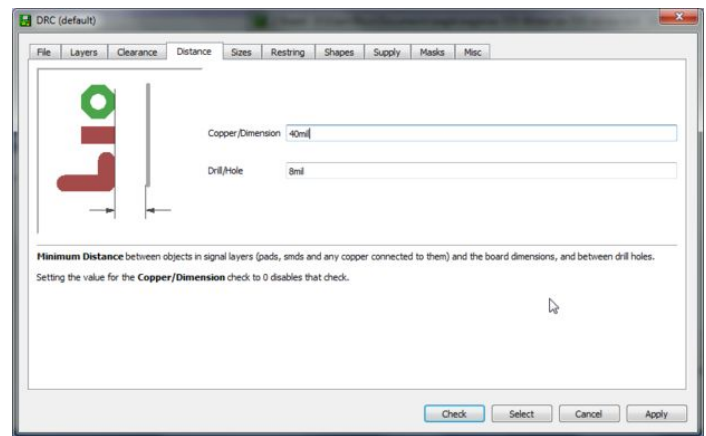
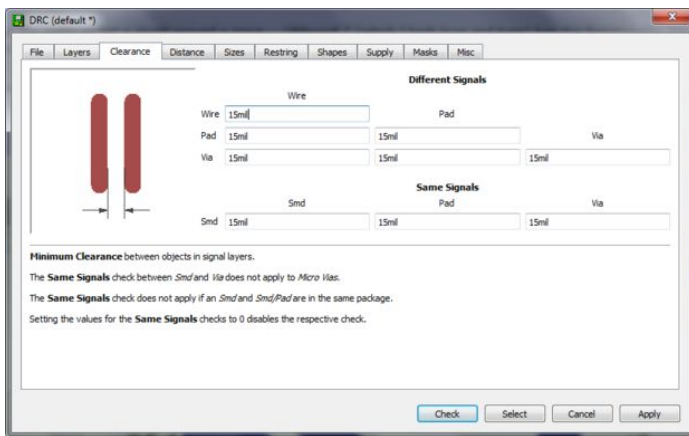
Gap factor for meanders for differential pairs - For high-speed signalling, you want both wires in a differential pair to be the exact same length. Depending on the routing, this may not be the case, so to make the shorter trace match the longer one, a 'meander' is put in. The Gap Factor adjusts the size of these meanders.

After you get all the settings the right way, click Check.

(after setting all of this up, you can just type 'drc' and hit enter twice to run the drc again)







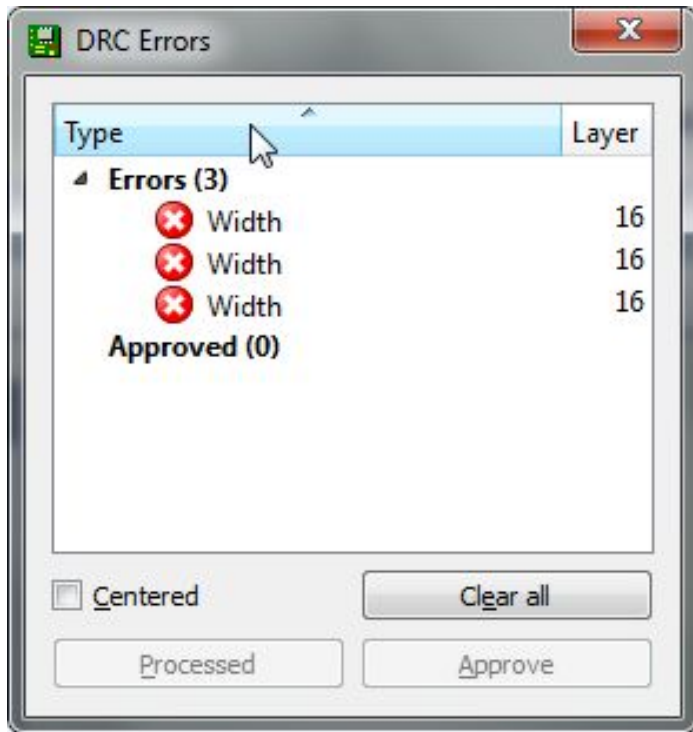


## Step 17: DRC Results

As you can see, I ran into three errors. Some of my traces weren't quite wide enough.

To fix trace width, there are two options:

- 1) Right-click on the trace, click Properties, and adjust the width
  - 2) Click on Change... (the wrench), click Width, and select a new width
- (alternatively, type 'Change width .024', or whatever you want to change, then click on the offending traces.)
  - - The Change tool is very powerful. Learn how to use it and your experience with Eagle will be much improved.



## Step 18: You're done

Once you've finished routing and have no more DRC errors, you're done with this tutorial. However, computer renderings are boring. To actually get your board made, here are a few resources:

### PCB Fabrication

Do it yourself - Really quick turnaround, not as accurate.

OSH Park - 2-week turnaround, I've used this and have received nothing but stellar boards. USA-based, free shipping, \$5/(3 copies of 1 sq in), \$10/(3 copies of 1 sq-in of four-layer board). Excellent deal for smaller boards, still competitive with some larger ones.

Advanced Circuits - More expensive, professional manufacturer based in Colorado. They do awesome work and offer sponsorships for student projects. The best deal that I can find on this one is \$33 each, where you buy a minimum of four PCBs of up to 60 sq-in and pay \$33 per.

Other fabrication services that I have heard of, but haven't used:

Sseed studio  
BatchPCB

### Parts:

Digikey  
Newark  
Mouser

Sparkfun - This is an awesome website that sells useful hard-to-find electronics project-related stuff, in addition to some more standard fare. While they don't have the greatest deals, they make up for it with good customer/community support and overall just being awesome.

eBay - If you want good deals on questionable parts from sketchy sellers, look no further. The robot team that I'm on managed to get five SICK laser range finders for \$300 total (and 3.5 of them actually worked!!! Typically one working one runs you a couple thousand)

## Step 19: Questions? Comments? Suggested Improvements?

The comments section is always open. Or, if you happen to be in the same room as me, just ask.

## Related Instructables



**Draw Electronic Schematics with CadSoft EAGLE**  
by westfw



**Making Printed circuit Boards at or Near Home: A Comprehensive Overview, Almost a Guide**  
by russ\_hensel



**Eagle-ize Leevonk's PIC protoboard** by westfw



**Turn your EAGLE schematic into a PCB** by westfw



**Render 3D images of your PCBs using Eagle3D and POV-Ray** by ongissim



**Make hobbyist PCBs with professional CAD tools by modifying "Design Rules"**  
by westfw